A NUMERICAL SIMULATION OF THE FLOW AND VORTEX STRUCTURES AROUND A SIMPLIFIED CAR MODEL

Hanis Rasyidah binti Abdullah[1], Intan Liyana binti Ramli[2] and Nurulhuda binti Khalid[3]

[1]Department of Mechanical Engineering, Politeknik Banting Selangor
hanis@polibanting.edu.my
intanliyana@polibanting.edu.my
nurulhuda_khalid@polibanting.edu.my

ABSTRACT. The ASMO model exhibits many of the 3D flow structures exhibited by real passenger cars. The scope of this work is an analysis of the applicability of the open-source CFD toolbox OpenFOAM for the prediction of external automobile aerodynamics using steady/unsteady RANS simulation as well as hybrid RANS-LES in combination with unstructured grids including up to 5×10^6 cells. This work is investigating the drag, lift, and pressure coefficient, flow field and flow structures in the rear section of the ASMO model using steady and unsteady RANS and hybrid RANS-LES. The results are validated by using experimental measurements by Daimler and Volvo. The flow structures around the ASMO model are constructed by using different vortex identification criteria. The inlet velocity is changed to know its influence. The results show that k−ω−SST gives the best for steady simulation and IDDES for unsteady simulation. IDDES can capture small vortices in the rear area of the ASMO model. Reduction of inlet velocity is affected the SLH simulation result because it is dependent on the flow separation. In conclusion, hybrid URANS-LES is the best model to study the aerodynamics of car models compared to URANS simulation.

KEYWORDS: ASMO model; SLH; OpenFOAM; automobile aerodynamics

1 INTRODUCTION

Aerodynamics design of the body the aerodynamics design of the car is the most critical aspect in car design (V.M.Lakshamaih, S. Srinivasarao, 2019). Aerodynamics is very essential for a car's performance, handling and stability. A low drag is a prerequisite for a fuel-economy car since the price of fuel becomes higher. Thus, drag is considered a prime concern in car design. Wolf Heinrich Hucho said that for a vehicle of speed 70km/hr and above, the prominence of wind is the major factor affecting fuel economy and stability (Hucho, 1998). So, a proper focus is needed in the aerodynamic design of a car. This study is done using 50m/s and 25m/s which is higher than 70km/hr (19.4m/s). Z M Saleh and A H Ali have investigated the drag reduction techniques in a car model. The study was performed at inlet velocity (V=10,20,30,40 m/s) (Ali, 2020).

The drag depends on the flow field around and through the car. The total drag of bluff bodies like a car is dominated by the pressure drag. Flow separation at the rear area is the main contributor to this pressure difference. It is caused by the geometry of the car. In this case, the Aerodynamiches Studien Modell (ASMO) car model is used as the geometry. The flow field in the recirculation area behind the ASMO car model is highly complex, unsteady and three-dimensional. Partial-scale or full-scale clay models are used in traditional aerodynamic development for experiments. These models need to include the car details to ensure geometric fidelity which is costly and time-consuming. The use of large wind tunnels requires planned scheduling and is expensive. In addition, certain desired data cannot be obtained by those models such as the vortices in the separation area.

Computer Fluid Dynamics (CFD) is a virtual testing bench which that can examine the aerodynamics of vehicles and may play a more important role in the future year due to the disadvantages of the experiment. This study is using OpenFOAM application. CFD with its ability can display the flow properties in detail and offer additional capability. Hucho said that the flow past a car is shown to be governed by different types of separations. Generally separated flow is time-dependent. However, due to time constraints, computer nowadays is not fast enough to compute flow by Direct Numerical Simulation (DNS) (Hucho, 2003). Even though the flow can calculate using Large Eddy

Simulation (LES), it will take a very long time. In this study, hybrid models which combine LES and RANS are used to produce the flow past an ASMO model to ensure good results and economically time-consuming.

2 LITERATURE REVIEW

Fluid flows are governed by partial differential equations (PDEs). These equations are representing conservation laws for the mass, momentum and energy. Computation techniques replace the PDE systems with a set of algebraic equation which can be solves using digital computer in CFD. CFD has three main component of software tools which are solvers, pre-processing and post-processing that is used to handle the object from start point till analysing the result (Thabet et al., 2018). However, the CFD simulation is not 100 percent reliable. The input data may come from guessing and imprecision. The mathematical model of the problem also maybe inadequate and the result accuracy is limited depending on computing power. Stability of numerical scheme, mesh, time step and stopping criteria are also affecting the quality of simulation. The turbulence model can be classified based on computational expense, which corresponds to the range scale. Finer resolution of simulation is needed to resolve more turbulence scale which needs higher computational cost. If a majority of turbulence scales are not resolved, the accuracy will decrease although it can reduce the computational cost.

Various models can provide different level of closure. RANS-based model is introducing Reynolds stresses which are second order tensor of unknowns. This model use 'timeaveraged' equation means that statically unsteady flows can equally be treated. RANS has been the backbone for the last few decades in modern CFD method for simulating the turbulent flow due to its less costing computing requirement and affordable to use (Yusof et al., 2020). K−ω models outperform the $k-\varepsilon$ model in terms of overall performance although its computational cost is higher than the $k-\varepsilon$ model (Kamal et al., 2021). Large Eddy Simulation (LES) is a model use by removing the smallest scales of the flow through filtering operation. Detached eddy simulation (DES) and other hybrid models use combination between RANS and LES. The RANS model switches to subgrid scale formulation in region where turbulence length scale is too fine for LES calculation. Direct numerical simulation (DNS) is extremely expensive because it resolves the entire range of turbulence length scale. DNS is very difficult for flow with complex geometries and flow configuration (Kornev, 2014). DNS is currently only can applicable for low Reynold number flow over simple geometry (Yusof et al., 2020). Zhang et al. have performed turbulence modelling effect on the CFD prediction of flow over a sedan vehicle. DES variants not showing superior performance over RANS models for predicting the absolute drag coefficient value, they were able to predict more complex flow structures with a higher fidelity (Zhang et al., 2019).

In the 90s, ASMO car model is created by Daimler Benz. This model is created with two main purposes which are to investigate low drag bodies in automotive aerodynamics and testing CFD codes with a neutral body not related to the development of an actual Mercedes car generation. Daimler-Benz and Volvo have made wind tunnel experiments for ASMO car models (I. Rodríguez et al., 2014). ASMO car model characteristics are smooth surface, square back rear, underbody diffuser, boat trailing and no pressure induces boundary layer separation (Perzon S. and Davidson L., 2000). Kamal et al. have used inlet velocity is approximately 50m/s to study aerodynamic influence on ASMO car model by using the Smagorinsky turbulence model (Kamal et al., 2021).

Kitoh et al. have performed RANS and LES simulations on the flow around a vehicle with a semi-complex underbody. By using the Reynolds number of 2.85×10^6, the velocity and pressure distribution are compared for LES and RANS. LES shows closer agreement with the experiment in pressure distribution along the top and bottom surfaces of the ASMO model (Kitoh et al., 2007). Tsubokura M et al. have validated the LES turbulence model for the assessment of vehicle aerodynamics. Firstly, they quantitatively validated the model on the ASMO car model using meshes 5.5×10^6 and 24.3×10^6 by comparing the mean pressure distribution on the vehicle surface between a conventional RANS and wind tunnel measurement. After that, the LES model is applied to full-scale vehicles with more complex geometry to qualitatively study the capability of capturing flow structure around the vehicle (Tsubokura M et al., 2009). Compared to the experimental result, LES models gives better result than RANS models in all studies. As conclusion, LES is a powerful tool for vehicle aerodynamic assessment which can provide precious aerodynamic data that cannot be provided by wind tunnel tests and RANS simulations.

3 METHODOLOGY

The computational domain for this case is a rectangular box of 2m×2m×9m. The front of the car is located 4m from the inlet boundary. The virtual model is set up as close to the physical model as possible. The first step in pre-processing is grid generation. Grid generation is a very critical step to get a good result. A good construct mesh can improve the quality of the solution. For this case, an unstructured mesh is generated in the computational domain around the ASMO car model. The grid generation is started by constructing the computational domain by using blockMesh utility. One refinement box is located around the ASMO car model and another is located at the turbulence location which is at the rear area of the ASMO model. The drag coefficient is used to compare 6 meshes with two experimental results from Volvo and Daimler. Meshes with 1.6 million cells and 3.8 million cells are chosen because they are near the experimental result.

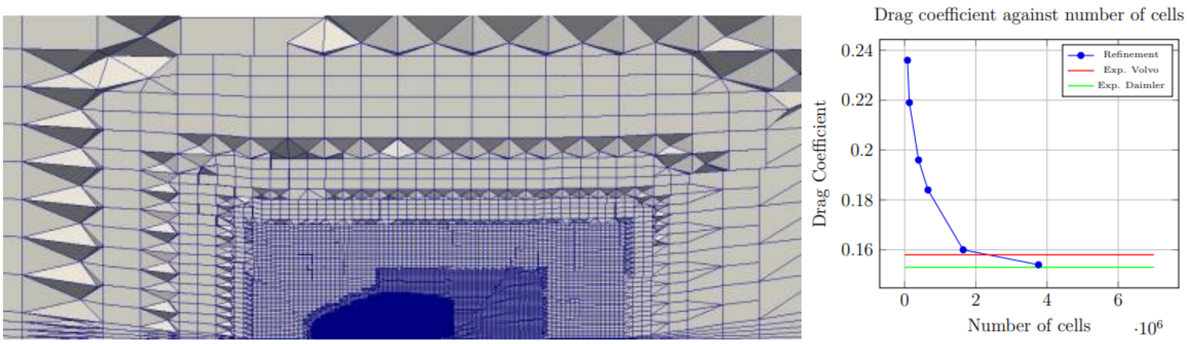


Figure 1: Mesh structures and drag coefficients comparison against number of cells.

The boundary is divided into five patches which are inlet, outlet, lowerWall, upperWall and frontAndBack. Some calculation is needed for initial boundary conditions which are turbulence kinetic energy, specific dissipation rate, turbulence dissipation rate and Reynolds Number. The initial and boundary conditions are set up for velocity and pressure parameters. For k−ω−SST, the initial value of U, p, k, ν_t and ω is used for the simulation. k−ε simulation need U, p, k, ν_t and ε. While the Spalart Allmaras and IDDES simulation use U, p, ν_t and $\tilde{\nu}$ and SLH needs U, p, ν_t , k and ω. So, parameters k, ν_t ω, $\tilde{\nu}$ and ε are set up for initial and boundary conditions. Then, a numerical setup needs to be done before the simulation started. SIMPLE is used for steady simulation and PIMPLE which is a combination between SIMPLE and PISO is used for k−ω−SST transient simulation. IDDES and SLH simulations use PISO for a solution. The setting in the fvSchemes and fvSolution is very important to make sure the simulations are stable, converge and get accurate results.

The transient simulation will not converge because it is in an unsteady state. For the transient simulations such as transient k−ω−SST, IDDES and SLH, steady-state simulations need to be run first. After it is stable, the simulation is continued with simulation in a transient state. For different velocity comparisons, 25ms^{-1} is used to compare with 50ms^{-1}. Setup for both velocities is run for three simulations which are transient k−ω−SST, IDDES and SLH.

After all simulation is done, post-processing is run to get the data at a certain point and to see the flow structure at a certain time. Pressure, velocity and turbulence kinetic energy at the latest time step have been extracted. The mean value of pressure and velocity are used for transient simulation. ParaView is used as a post-processing tool that includes a reader module to read data in OpenFOAM format. The solutions that calculate in OpenFOAM simulation enable it to be visualized in ParaView. The structure of vortices around the ASMO car model is presented by using different vortex identification criteria which are Q and Lambda2. Paraview is used to see the structure by using a contour filter. It is important to know the LES and RANS regions in the hybrid simulation to analyse the simulation. An object function name DES Model Regions is used to differentiate the LES and RANS regions in IDDES simulation. The SLH simulation is automatically produced RANS files and the files can show the RANS region in Paraview.

4 RESULT AND DISCUSSION

This study shows the lift and drag coefficient. Experimental values from Daimler and Volvo are used as references and shown in table 1. Although the drag coefficient for a steady state is lower than the transient state it is less accurate because it ignores many of the cross terms and higher-order terms dealing with time. For steady simulation, the k−ω−SST simulation results for the drag coefficient are nearest to the experimental values. In the transient state, IDDES simulation shows the best result which is nearest to the experimental value. The drag coefficient for IDDES simulation is far lower than k−ω−SST and SLH simulations for inlet velocity 50m/s. However, the IDDES and SLH results are nearly the same for an inlet velocity of 25m/s. Since the pressure drag coefficient is nearly the same for all transient state, this indicates that the friction drag coefficient is the main cause of the differences. IDDES simulation uses WMLES which is activated if the inlet is turbulent. This function can reduce log layer mismatch and produce accurate results. This function is changed depending on the inlet flow. SLH function uses the shield function. This function makes the RANS/LES interface is forced to move farther from the wall. k−ω−SST simulation has a y+ mean value of 19 at the buffer zone. In the buffer zone, the result cannot be calculated accurately. IDDES and SLH simulations are hybrid simulations that combine RANS and LES models. The LES model is produced accurate results because it can capture smaller vortices compared to the RANS model.

Table 1: Lift and Drag coefficient.

State	Velocity	Simulation	C_d(Pressure)	C_d(Friction)	C_d	C_l
Steady	50m/s	k-ω-SST (Coarse)	0.121 (76.1)	0.038 (23.9)	0.159	-0.0588
		k-ω-SST (Fine)	0.115 (74.6)	0.039 (25.4)	0.154	-0.0763
		$k-\varepsilon$	0.116 (72.8)	0.044 (27.2)	0.160	0.0015
		SA	0.134 (90.6)	0.014 (9.4)	0.148	-0.0589
Transient	50m/s	k-ω-SST	0.142 (77.4)	0.041 (22.6)	0.183	-0.0013
		IDDES	0.147 (90.7)	0.015 (9.3)	0.162	-0.0806
		SLH	0.146 (79.1)	0.039 (20.9)	0.185	-0.0487
	25m/s	k-ω-SST	0.143 (77.6)	0.042 (22.4)	0.185	-0.0399
		IDDES	0.144 (86.9)	0.022 (13.1)	0.166	-0.0723
		SLH	0.143 (86.4)	0.023 (13.6)	0.166	-0.0794
		(Daimler)			0.153	
		(Volvo)			0.158	

Pressure is the most crucial parameter to analyse in this simulation since the drag is produced mainly from the pressure drag where the pressure drag is higher than friction drag. It also can be compared with the experimental results. The pressure coefficient at the surface of the ASMO body is taken along the profile of the symmetry plane. They are divided into four areas which are base, frontal, underbody and over the roof. Figure 2 shows the results of base areas. IDDES simulation result for velocity 50m/s is nearly the same as the k−ω−SST simulation. They are nearly the same as the result from Volvo's experiment. SLH simulation result is nearly the same as Daimler's experiment which is lower than other simulations. For velocity 25m/s shows the pressure coefficient of SLH simulation same as IDDES simulation and follows the experimental result from Daimler. The SLH simulation result is far from the IDDES simulation result for inlet velocity 50m/s but nearly the same as the IDDES simulation result for inlet velocity 25m/s maybe because of the flow. The flow with higher velocity will have a higher Reynolds number and more turbulence.

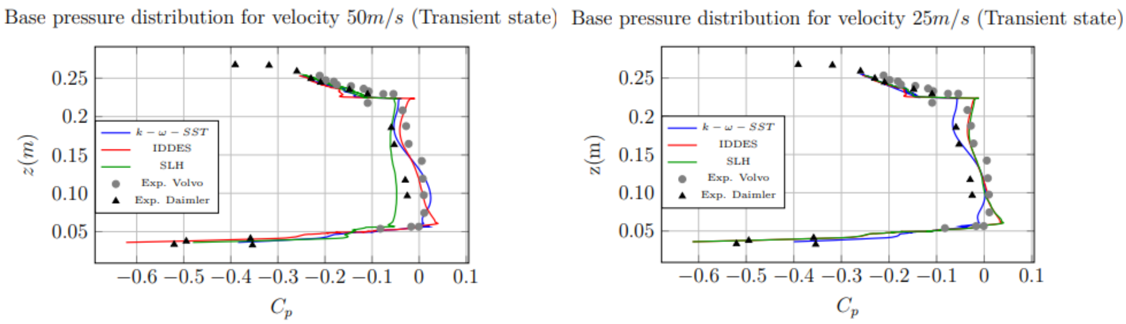
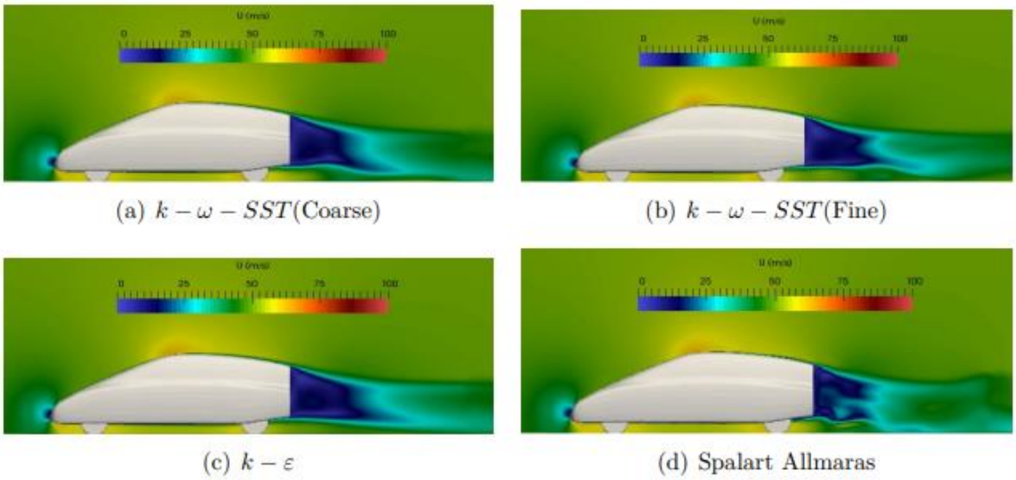


Figure 2: Pressure coefficient in the rear section

The rear area of the ASMO body also has low velocity. Low-velocity area has negative velocity caused by turbulence. Figure 3 shows that Spalart Allmaras simulation produces unstable velocity rear area of the ASMO body where turbulence happens compare to other simulations. Spalart Allmaras model is not as suitable for turbulence areas. k−ε simulation produces a smooth velocity field at the rear area compared to other steady-state simulations because this model is suitable at the turbulence area. This model produces nearly the same result as the transient state simulations. The results in transient simulations are nearly same for all simulations.

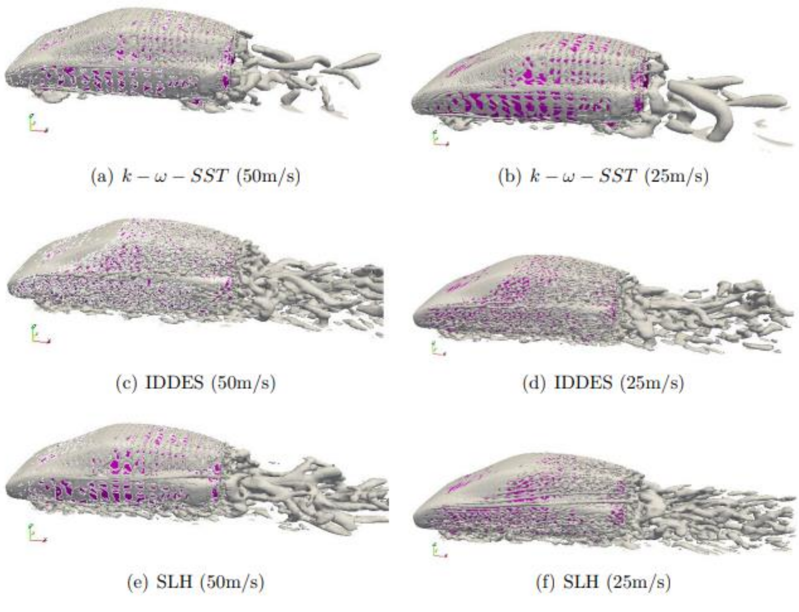


(a) $k-\omega-SST$(Coarse) (b) $k-\omega-SST$(Fine)

(c) $k-\varepsilon$ (d) Spalart Allmaras

Figure 3: Velocity around ASMO car model (Static State)

The smooth isosurface in front of the body shows laminar flow. After the highest point of the car, where the pressure is reduced, the small vortices structures are lifted away from the ASMO body. A lot of big and small vortices produce in the rear of the ASMO body where recirculation occurs. At the tire area, the vortices can be seen. In a k−ω−SST simulation, only a few the vortices can be captured compared to IDDES and SLH simulations which produce a less accurate drag coefficient. The difference between inlet velocity 25m/s and 50m/s of SLH simulation can be seen. SLH simulation with an inlet velocity of 25m/s produces nearly the same vortices structures compare to IDDES simulation. That's why their drag coefficient is the same. While the inlet velocity of 50m/s captures only big vortices without capturing the small vortices in the rear area of the ASMO body producing less accurate results.



(a) $k-\omega-SST$ (50m/s) (b) $k-\omega-SST$ (25m/s)

(c) IDDES (50m/s) (d) IDDES (25m/s)

(e) SLH (50m/s) (f) SLH (25m/s)

Figure 4: Vortex identification criteria (Q)

5 CONCLUSION

K−ω−SST gives the best for steady simulation but the worst result for unsteady simulation. IDDES simulation gives the best result compared to the experimental value. It is predictable because k−ω–SST simulation only uses the RANS model. SLH simulation results depend on inlet velocity, with low inlet velocity, the result is nearly the same as with IDDES simulation but for high inlet velocity, the result is near to k−ω−SST simulation. This phenomenon happens due to the turbulence area of the SLH simulation solved by using the RANS model. Differing from IDDES simulation, the turbulence area is solved by the LES model. The LES model produces a more accurate result compared to the RANS model because it can capture smaller vortices. The turbulence area is presented by recirculation at the rear area after the ASMO body.

This study only compares two types of hybrids which use different RANS model where SLH simulation uses k−ω−SST and IDDES simulation uses Spalart Allmaras model. It is better to compare with a hybrid simulation that uses the same RANS model such as IDDES simulation using k−ω−SST for the RANS model. The SLH simulation also needs improvement so that it can solve the turbulence area with the LES model. The ASMO car model is very interesting to study because its shape is nearly the same as a real passenger car. The drag coefficient influenced by the angle of the diffuser is fascinating to investigate. The straight part at the rear area of the ASMO body produces a lot of drag force. If the rear shape of the ASMO body is changed to have a small slope, the positive effect can be studied in further work.

6 REFERENCES

Ali, Z. M. (2020). Numerical Investigation of Drag Reduction Techniques in a . *3rd International Conference on Engineering Sciences* (p. 671). IOP Publishing.

Hucho, W.-H. (1998). *Aerodynamics of Road Vehicles.* Butterworth: Boston.

Hucho, W.-H. (2003). Aerodynamics of road vehicles-a challenge for computational. *European Automotive CFD Conference.*

I. Rodríguez et al. (2014). Flow and turbulent structures around simplified car models. *Computer & Fluids*, 96:122–135.

Kamal et al. (2021). A Review of Aerodynamics Influence on Various Car Model Geometry through CFD Techniques. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, 109-125.

Kitoh et al. (2007). Large eddy simulation on the underbody flow of the vehicle with semi-complex underbody configuration. *World Congress,*, (pp. 2007-01-0103). Detroit, Michigan,.

Kornev, N. (2014). *Lectures on Computational Fluid Dynamics and Heat Transfer with Applications.* Rostock, Germany: bookboon.com.

Perzon S. and Davidson L. (2000). On transient modeling of the flow around vehicles. *ACFD 2000*, (pp. 720–777). Beijing, China.

Thabet et al. (2018). Computational Fluid Dynamics: Science of the Future. *International Journal of Research and Engineering*, Vol.5, No. 6, pp. 430-433.

Tsubokura M et al. (2009). Computational visualization of unsteady flow around vehicles using. *Computer & Fluids*, (pp. 38:981–990).

V.M.Lakshamaih, S. Srinivasarao. (2019). CFD Research on Car Body. *International Journal of Recent Technology and Engineering (IJRTE)*, ISSN: 2277-3878, Volume-8, Issue-2S3.

Yusof et al. (2020). A Short Review on RANS Turbulence Models. *CFD Letters 12*, 83-96.

Zhang et al. (2019). Turbulence Modeling Effects on the CFD Predictions of Flow over a Detailed Full-Scale Sedan Vehicle. *Turbulence and Transitional Modeling of Aerodynamic Flows*, 4(3).